Tutorial: Dam-Break Simulation Using FLUENT*'s Volume of Fluid Model*

Purpose

This tutorial examines the dam-break problem using the Volume of Fluid (VOF) multiphase model.

This tutorial demonstrates how to do the following:

- Set up a dam-break problem.
- Choose the time step by estimating the maximum possible velocity of the interface and the grid cell dimension.
- Solve the problem using the VOF model.
- Manipulate the solution parameters.

Prerequisites

This tutorial assumes that you are familiar with the FLUENT interface and that you have a good understanding of basic setup and solution procedures. In this tutorial, you will use VOF multiphase model, so you should have some experience with it. This tutorial will not cover the mechanics of using this model; instead, it will focus on the application of this model to solve a dam-break problem.

If you have not used this model before, it would be helpful to first refer to the FLUENT 6.3 User's Guide and the FLUENT 6.2 Tutorial Guide.

Problem Description

The initial setup of the dam-break problem is shown in Figure 1.

In this problem, a rectangular column of water, in hydrostatic equilibrium, is confined between two walls. Gravity is acting downwards with a magnitude of -9.81 m/s^2. At the beginning of the calculation, the right wall is removed and the water is allowed to flow out to the horizontal wall.

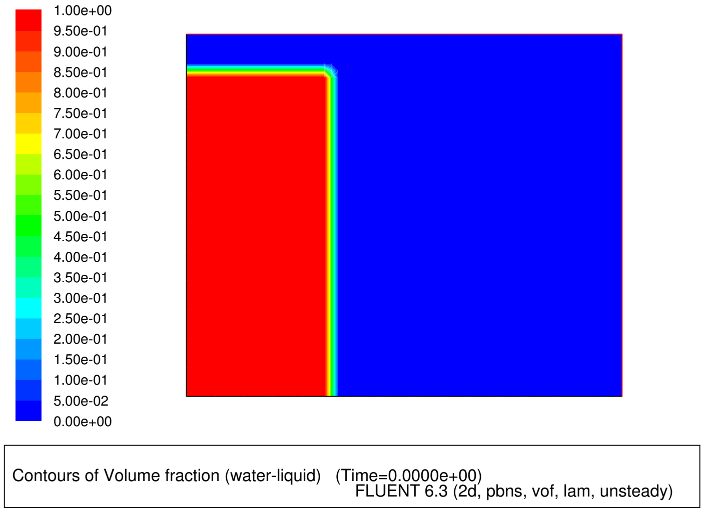

Figure 1: Initial Setup of the Problem

Preparation

1. Copy the mesh file, `dambreak.msh.gz` to the working folder.
2. Start the 2D (2d) version of FLUENT.

Setup and Solution

Step 1: Grid

1. Read the grid file `dambreak.msh.gz`.
2. Check the grid.

   Grid ⟶Check
3. Display the grid (Figure 2).

   Display ⟶Grid...

   (a) Click the Colors... button to open the Grid Colors panel.

      i. Select Color By ID.

         This will assign a different color to each zone in the domain, rather than to each type of zone.

      ii. Close the Grid Colors panel.

   (b) Click Display.

   (c) Close the Grid Display panel.

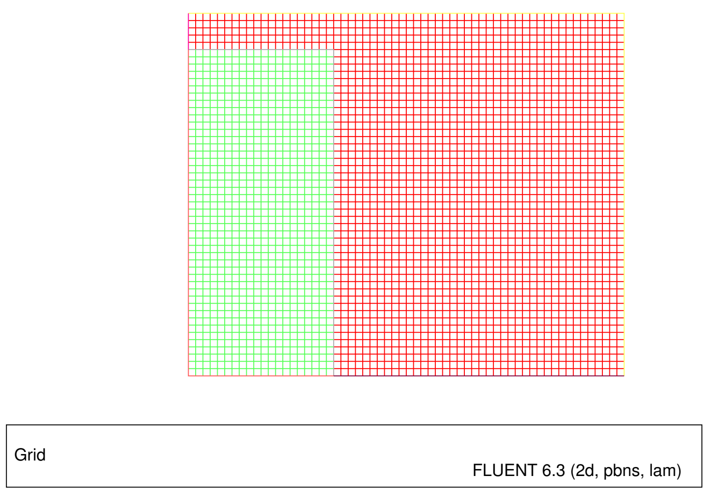

Figure 2: Grid Display

Step 2: Models

1. Define the solver settings.

   Define ⟶ Models ⟶ Solver...

   (a) Select Unsteady for Time.

   (b) Click OK to close the Solver panel.

2. Define the model settings.

   Define ⟶ Models ⟶ Multiphase...

   (a) Select the Volume of Fluid multiphase model.

   (b) Enable Implicit Body Force Formulation.

   (c) Click OK to close the Multiphase Model panel.

Step 3: Materials

The default properties of water and air are the correct values for this problem. You can check them in the Materials *panel.*

1. Define the material properties.

   Define ⟶ Materials...

   (a) Retain the default settings for air.

   (b) Define the new material by copying water-liquid (h20<l>) from the Fluent Database.

   (c) Click Change/Create and close the Materials panel.

2. Define the primary (air) and secondary (water-liquid) phases.

   Define ⟶ Phases...

   (a) Specify air as the primary phase.

   (b) Specify water-liquid as the secondary phase.

   (c) Close the Phases panel.

Step 4: Operating Conditions

Define ⟶Operating Conditions...

1. Enable Gravity.
2. Enter -9.81 m/s^2 for Gravitational Acceleration in the Y direction.
3. Enable Specified Operating Density and retain the default value for Operating Density.
4. Click OK to close the Operating Conditions panel.

Step 5: Boundary Conditions

Define ⟶Boundary Conditions...

1. Set the boundary conditions for poutlet.
   (a) In the drop-down list for Phase, select water-liquid and click the Set... button.
      i. Retain the default value of 0 for Backflow Volume Fraction.
      ii. Click OK to close the Pressure Outlet panel.
   (b) Close the Boundary Conditions panel.

Step 6: Solution

1. Set the solution parameters.
   Solve ⟶ Controls ⟶Solution...
   (a) Enter 0.9 for Pressure, 0.7 for Momentum, and 1 for the remaining parameters in the Under-Relaxation Factors group box.
   (b) Select PRESTO! from the Pressure drop-down list, and First Order Upwind from the Momentum drop-down list in the Discretization group box.
   (c) Select PISO from the Pressure-Velocity Coupling drop-down list.
   PISO is recommended for transient flow simulations.
   (d) Click OK to close the Solution Controls panel.
2. Set the termination criteria for Pressure.
   Solve ⟶ Controls ⟶Multigrid...
   (a) Enter 0.001 for Termination Value for Pressure.
   (b) Click OK to close the Multigrid Controls panel.

3. Initialize the solution.

   Solve ⟶ Initialize ⟶ Initialize...

   (a) Retain the default values for all components and click Init.

   (b) Close the Solution Initialization panel.

4. Patch the initial distribution of the water-liquid.

   Solve ⟶ Initialize ⟶ Patch...

   (a) In the drop-down list for Phase, select water-liquid.

   (b) In the Variable list, select Volume Fraction.

   (c) Select water in the Zones To Patch list.

   (d) Set the Value to `1`.

   (e) Click Patch and close the Patch panel.

5. Enable the plotting of residuals during the calculation.

   Solve ⟶ Monitors ⟶ Residual...

   (a) Enable Plot.

   (b) Click OK to close the Residual Monitors panel.

6. Calculate the time step by estimating the maximum possible velocity of the interface and the grid cell dimension.

$$Courant = \frac{\triangle t}{\triangle x_{cell}} v_{fluid}$$

$$\rho g h = \frac{\rho}{2} v_{fluid}^2$$

$$v_{fluid} = \sqrt{2gh} \approx 10 m/s$$

$$\triangle t = \triangle x_{cell} / v_{fluid} \approx 0.01 sec$$

7. Set the time stepping parameters.

   Solve ⟶ Iterate...

   (a) Enter `0.01` (s) for Time Step Size.

   (b) Enter 20 for Number of Time Steps.

   (c) Enter 40 for Max. Iterations per Time Step.

   (d) Click Apply.

8. Save the initial case and data files, (`dambreak.cas.gz` and `dambreak.dat.gz`).

9. Click Iterate to start the calculation.

10. After FLUENT completes the specified number of iterations, run the calculation further successively specifying 30, 30, and 20 (a total of 80) for Number of Time Steps.

11. Save the case and data files after each set of iterations.

    This will help in viewing the progress of the VOF simulation.

Step 7: Postprocessing

The velocity vectors and contours of velocity for the VOF model are displayed for four time step values. The following figures (Figure 3 to Figure 9) show the progression of the solution as the number of time steps is increased.

1. Display velocity vectors after 20 time steps (Figure 3).
   (a) Select Velocity from the Vectors Of drop-down list.
   (b) Select mixture from the Phase drop-down list.
   (c) Select Velocity... and Velocity Magnitude from the Color By drop-down lists and click Display.
2. Display filled contours of volume fraction of water-liquid after 20 time steps (Figure 4).
   (a) Select Phases... and Volume Fraction from the Contours Of drop-down lists.
   (b) Select water-liquid from the Phase drop-down list and click Display.
3. Similarly, display the contours of volume fraction and velocity vectors after 50, 80, and 100 time steps (Figure 5 to Figure 10).

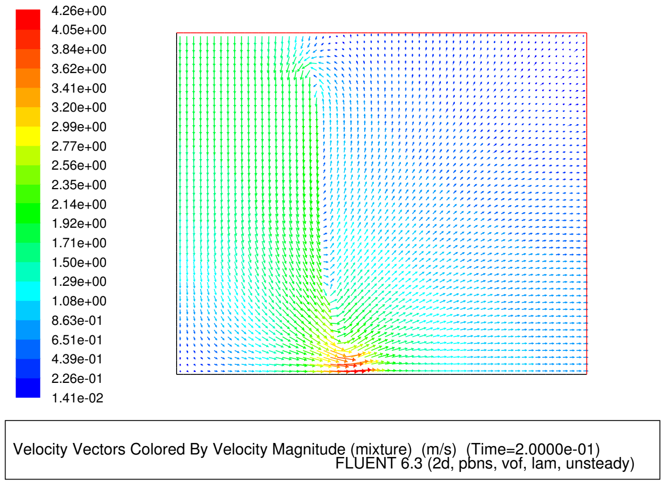

Figure 3: Velocity Vectors Colored By Velocity Magnitude after 20 Time Steps

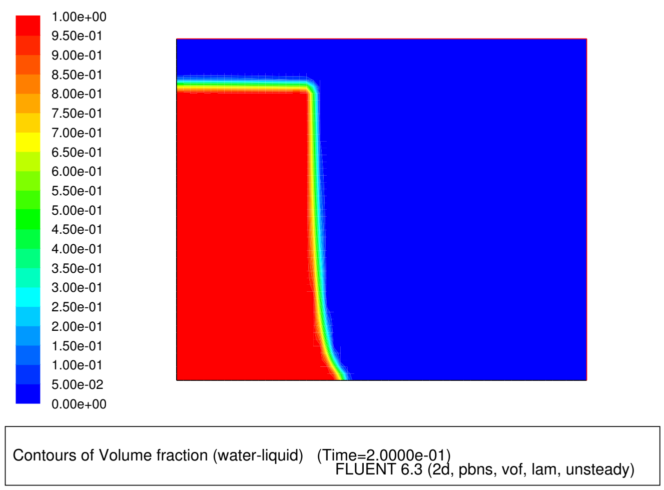

Figure 4: Contours of Water Volume Fraction after 20 Time Steps

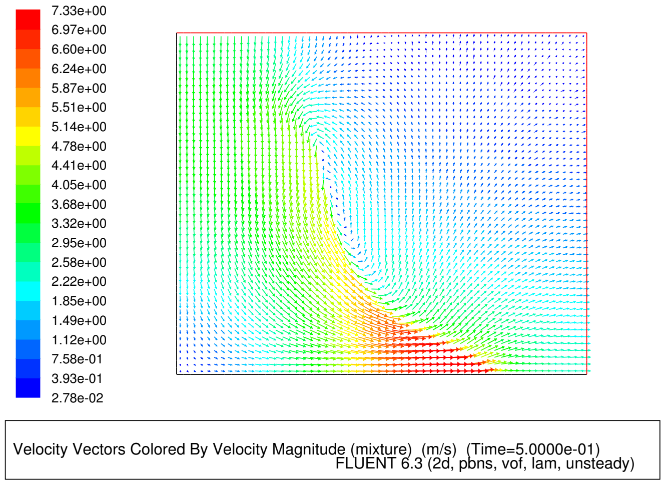

Figure 5: Velocity Vectors Colored By Velocity Magnitude after 50 Time Steps

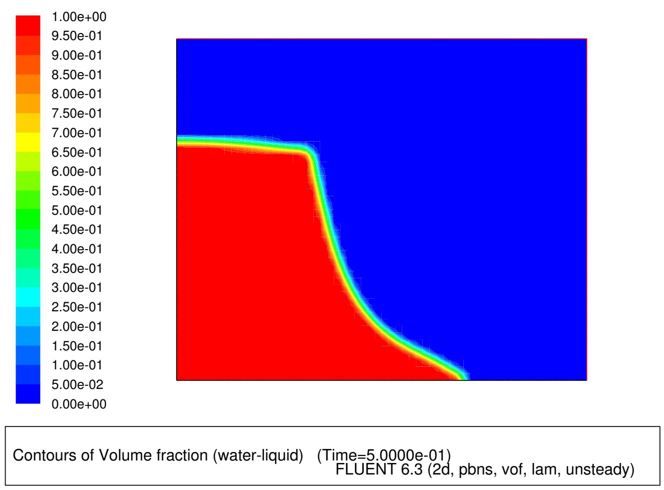

Figure 6: Contours of Water Volume Fraction after 50 Time Steps

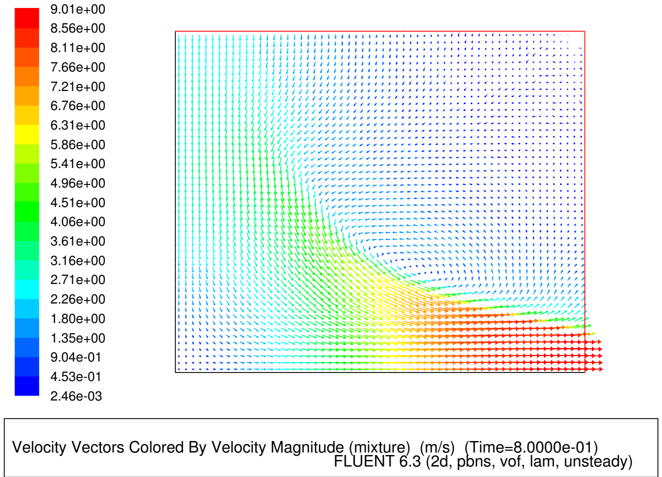

Figure 7: Velocity Vectors Colored By Velocity Magnitude after 80 Time Steps

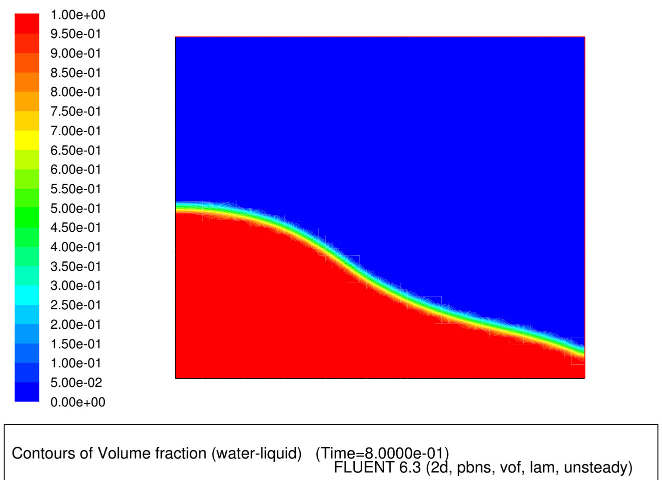

Figure 8: Contours of Water Volume Fraction after 80 Time Steps

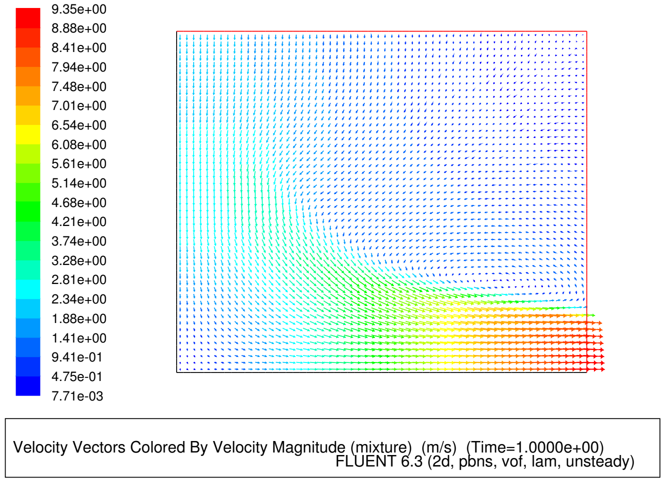

Figure 9: Velocity Vectors Colored By Velocity Magnitude after 100 Time Steps

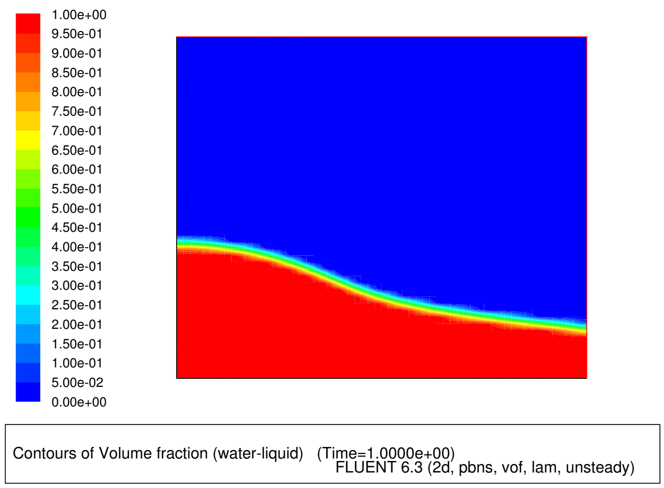

Figure 10: Contours of Water Volume Fraction after 100 Time Steps

Step 8: Manipulation of Solution Parameters

1. Solve the problem by manipulating different solution parameters.
   - Interface tracking scheme.
   - Enable/Disable PISO.
   - Pressure interpolation scheme.
   - Discretization scheme for momentum, volume fraction.
   - Reference density value and location.
   - Time step size.

Summary

In this tutorial, a dam-break problem was solved using the Volume of Fluid (VOF) model in FLUENT.